

Hot Forming Training Tutorial

Version 5.9.2.1



Engineering Technology Associates, Inc.
1133 E. Maple Road, Suite 200
Troy, MI 48083

Tel: +1 (248) 729 3010
Fax: +1 (248) 729 3020
Email: support@eta.com

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/DYNAFORM are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of the respective owners.

FOREWORD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

OVERVIEW

The eta/DYNAFORM software package consists of four programs. These programs represent the pre-processor, solver and post-processor. They are: eta/DYNAFORM, eta/Job Submitter, eta/POST and eta/3DPlayer.

eta/DYNAFORM is the pre-processor portion of the software package, which is used to construct the sheet metal forming models. It includes VDA and IGES translators for importing line data and a complete array of tools for altering or constructing line data and meshing it.

LS-DYNA is the software package's solver. eta/DYNAFORM has a complete LS-DYNA interface allowing the user to run LS-DYNA from eta/DYNAFORM.

eta/POST and eta/GRAPH are the post-processing portions of the package. These programs are used to post-process the LS-DYNA result files from the analysis. eta/POST creates contour, deformation, FLD, and stress plots and animations with the result files. eta/GRAPH contains functions for graphically interpreting the same results.

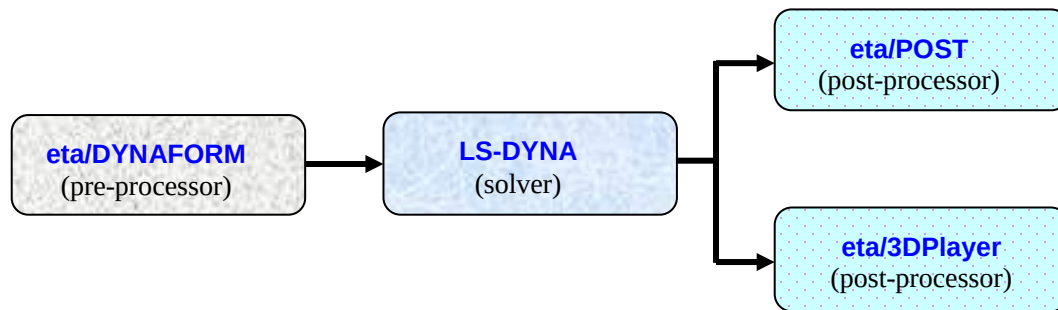


Figure 1: Components of eta/DYNAFORM solution package.

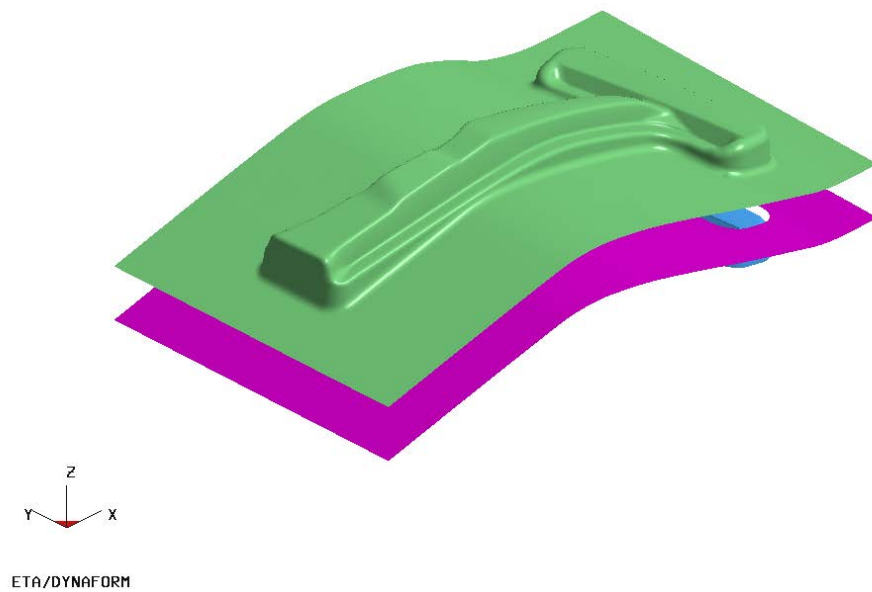
Each of the software components has its own manual which should be referenced for further information on running these programs. These manuals are:

eta/DYNAFORM Application Manual	A comprehensive training manual for using the eta/DYNAFORM software package for various applications.
eta/DYNAFORM User's Manual	A reference guide to the functions contained in the eta/DYNAFORM program (pre-processor).
LS-DYNA Keyword User's Manual	A reference guide to the LS-DYNA program (solver).
eta/POST User's Manual	A reference guide to the functions contained in the eta/POST program and eta/GRAPH program (post-processor).

TABLE OF CONTENTS

FOREWORD.....	I
OVERVIEW	II
TABLE OF CONTENTS.....	III
Hot Forming Analysis - B Pillar	1
Create an eta/DYNAFORM Database	2
I. Start eta/DYNAFORM 5.9.2.1.....	2
II. Open the Database.....	2
HOT FORMING SETUP	4
I. New Hot Forming Setup.....	4
II. General	5
III. Blank Definition	5
IV. Blank Material and Property Definition	8
V. Tools Definition.....	10
VI. Tools Material Definition.....	14
VII. Tools Positioning	15
VIII. Process Definition	18
IX. Control Parameters	20
X. Animation.....	21
XI. Gravity Loading.....	23
XII. Press Hardening.....	25
XIII. Submit Job	27
POST PROCESSING (with eta/POST).....	29
I. Reading the Results File into the Post Processor.....	29
II. Deformation	30
III. Thinning.....	31
IV. View Temperature Distribution	32
MORE ABOUT eta/DYNAFORM 5.9.2.1	34
CONCLUSION	35

Hot Forming Analysis - B Pillar



Create an eta/DYNAFORM Database

I. Start eta/DYNAFORM 5.9.2.1

For workstation/Linux users, enter the command “**df5921**” (default) from a UNIX shell to start eta/DYNAFORM5.9.2.1. For PC users, double click the eta/DYNAFORM5.9.2.1 (DF5921) icon from the desktop or choose eta/DYNAFORM from the program group.

After starting eta/DYNAFORM, a default database **Untitled.df** is created. The user needs to import CAD or CAE model to the database to start working.

II. Open the Database

From the menu bar, select **File→Open** to open the Open File dialog box, as illustrated in Figure 1.1. Select the database file **HotForming.df** and click Open to display the model illustrated in Figure 1.2.

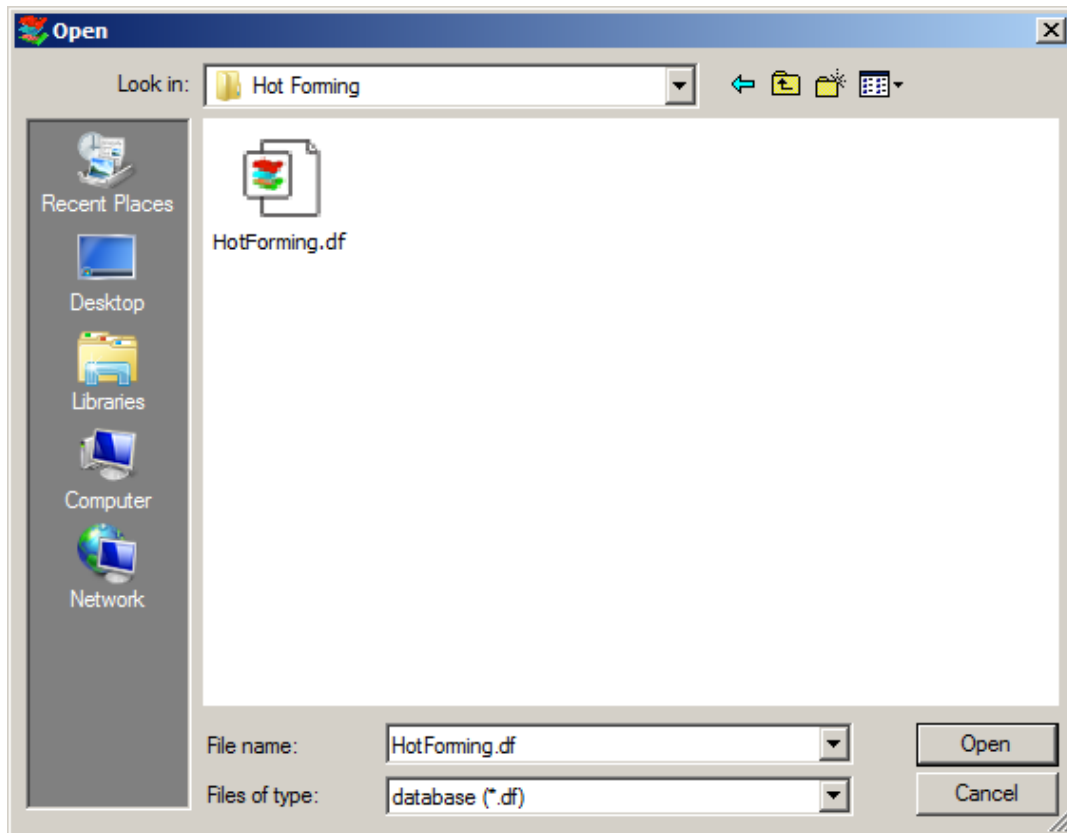


Figure 1.1 Open file dialog box

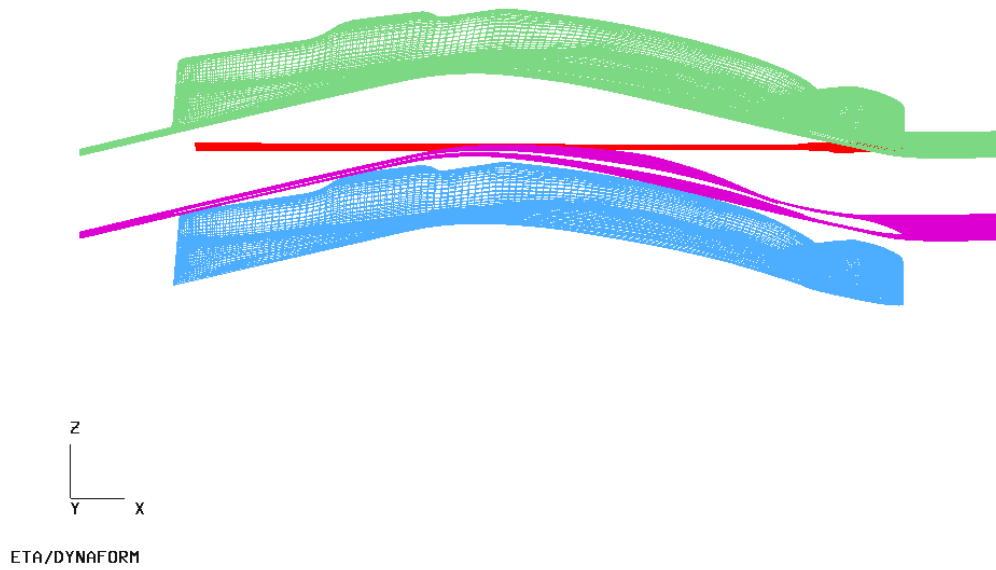


Figure 1.2 Opened Model

HOT FORMING SETUP

After finishing the preparation of model, click the **Hot Forming** from **AutoSetup** menu to display Hot Forming Setup.

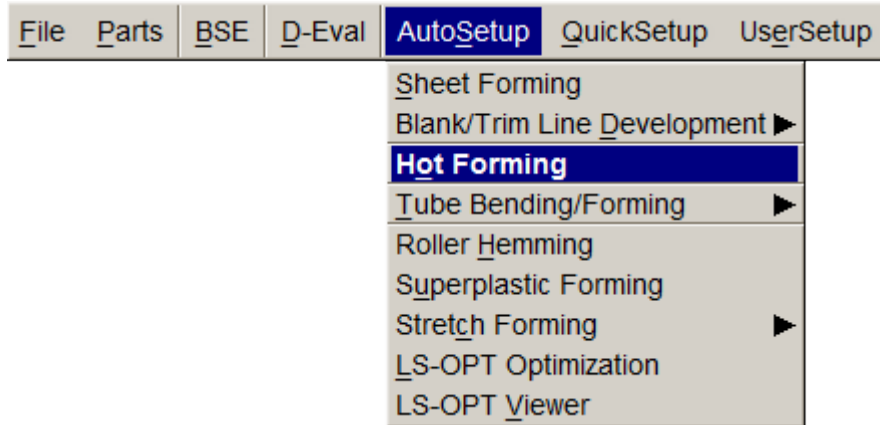


Figure 1.3 AutoSetup menu

I. New Hot Forming Setup

Clicking Hot Forming menu, the New Hot Forming dialog box appears for the user to define the basic setup parameters, as illustrated Figure 1.4.

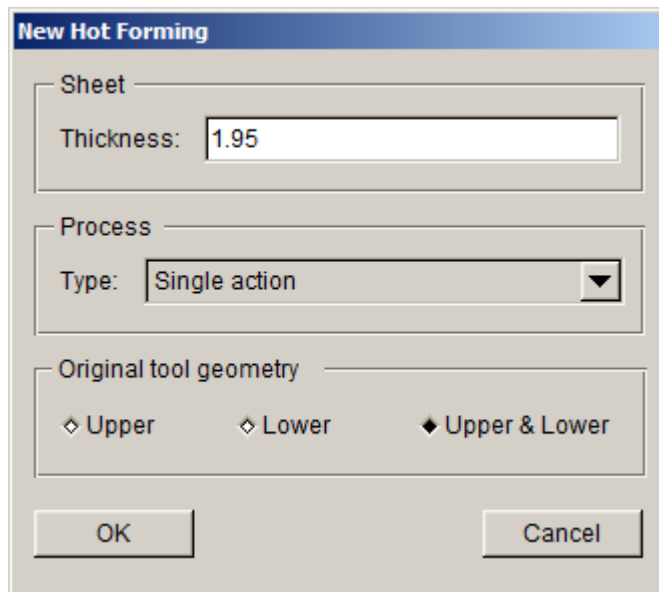


Figure 1.4 Sheet Forming Setup dialog box

1. Input blank thickness:**1.95** (mm).
2. Select process type: **Single Action**.
3. Select original die face: **Upper &Lower**.
4. Click **OK** to display the Hot Forming interface.

II. General

After entering **General** interface, the program will automatically create three stages: Gravity, Forming and Hardening. As the tool defined in the forming stage is cited in the gravity and hardening stages, the user is allowed to define the blank and tool from the forming stage, as illustrated in Figure 1.5.

You do not need to modify any parameters except for changing the **Title** into **Forming**. See Figure 1.5.

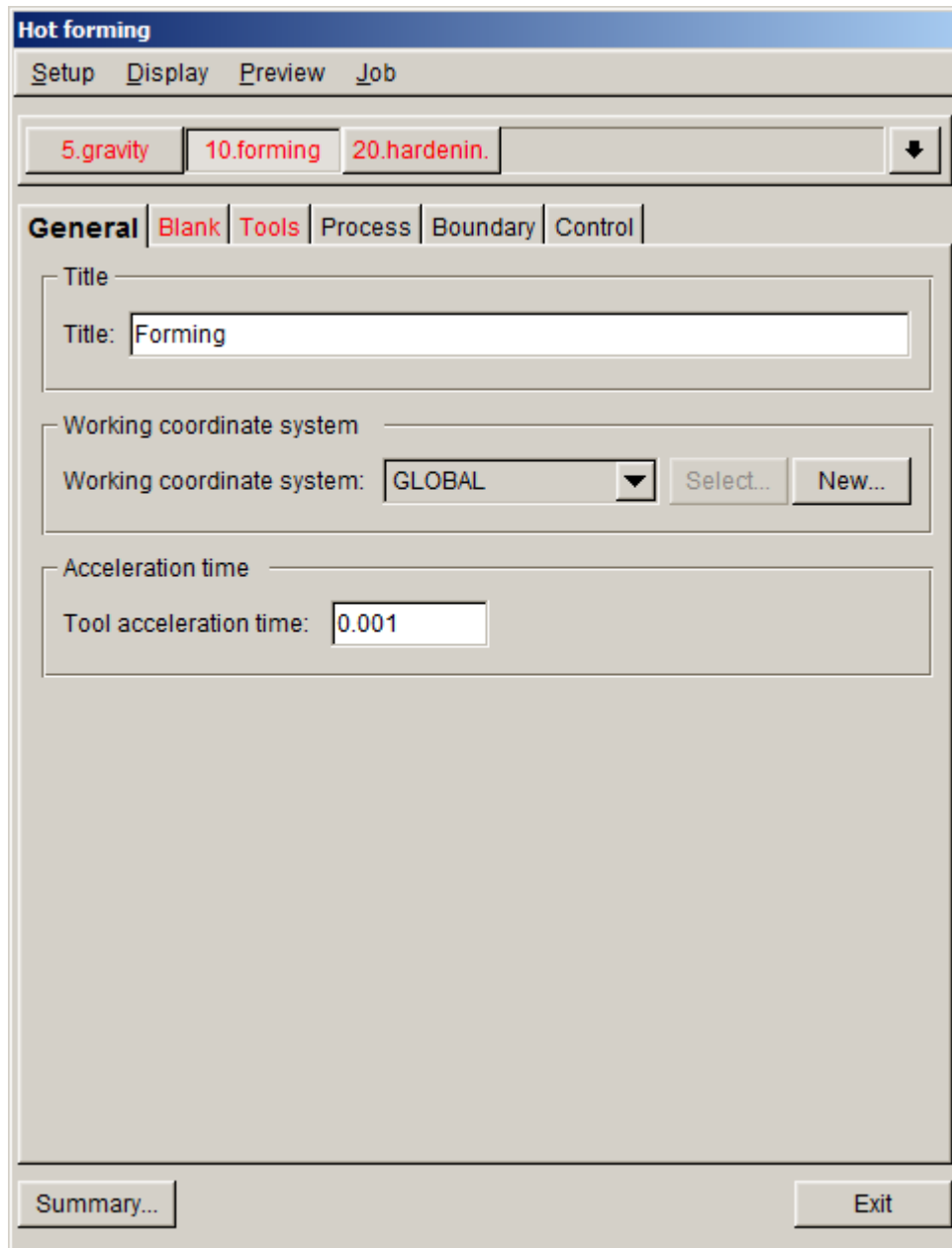


Figure 1.5 General interface

III. Blank Definition

1. Enter the Blank interface, and click the **Blank** tab to display the Blank Definition interface.

- Then, click the **Define geometry** button from **Geometry** field in the blank definition interface. See Figure 1.6.

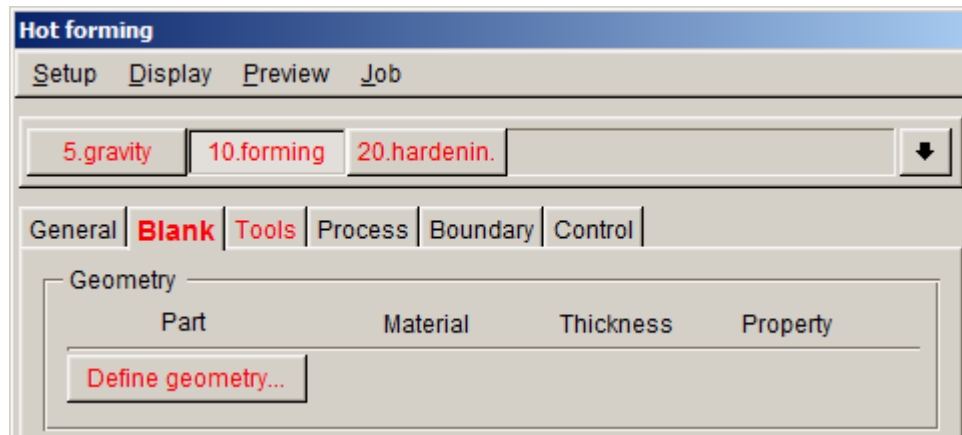


Figure 1.6 Define blank

- The Blank Generator dialog box illustrated in Figure 1.7 is displayed.

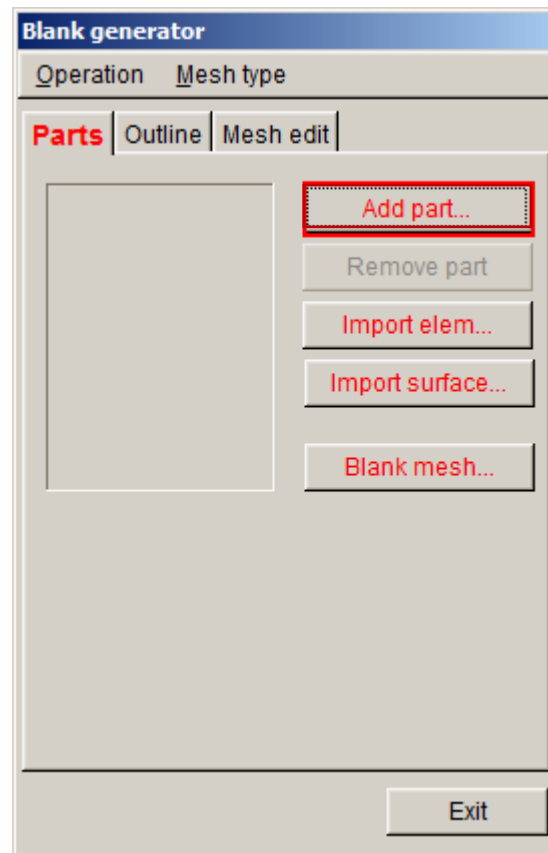


Figure 1.7 Blank generator dialog box

- Click the **Add Part** button to select the **BLANK** part from the SELECT PART dialog box illustrated in Figure 1.8.
- After selecting the part, click **OK** button to return to the Blank Generator dialog box. As illustrated in Figure 1.9, the BLANK part has been added to the list of blank.
- Click the **Exit** button to return to the blank definition interface. Now, the color of Blank tab

is changed from RED to BLACK. See Figure 1.10.

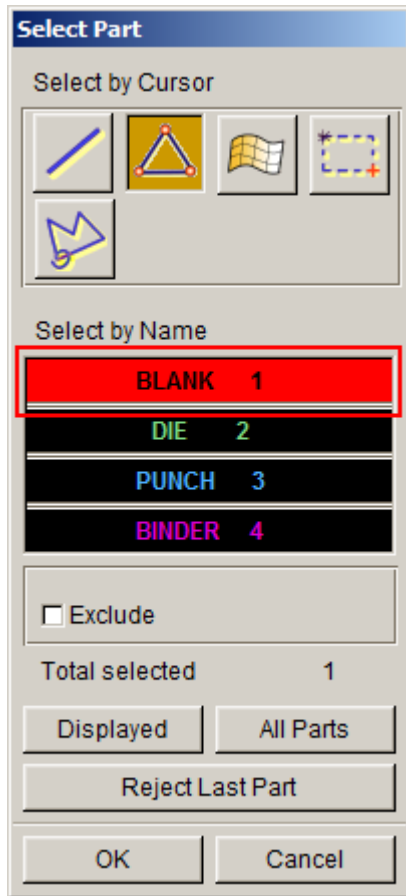


Figure 1.8 Select Part dialog box

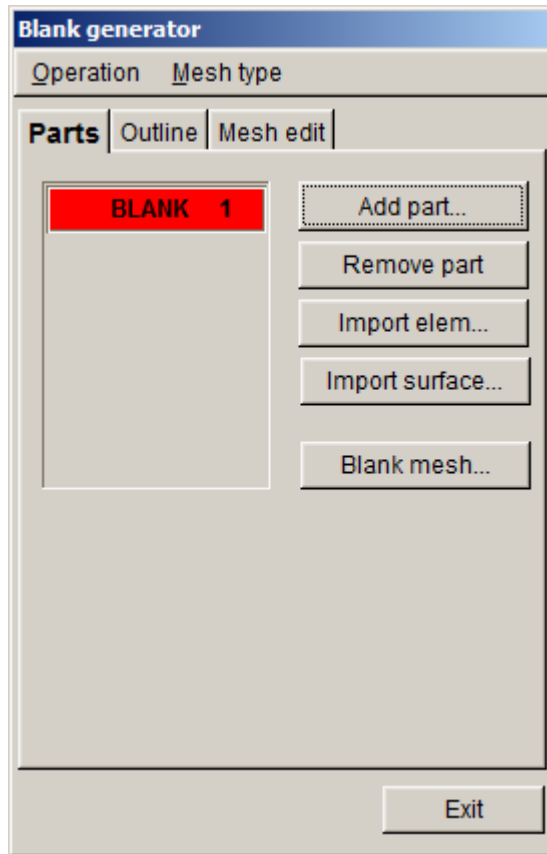


Figure 1.9 Blank generator

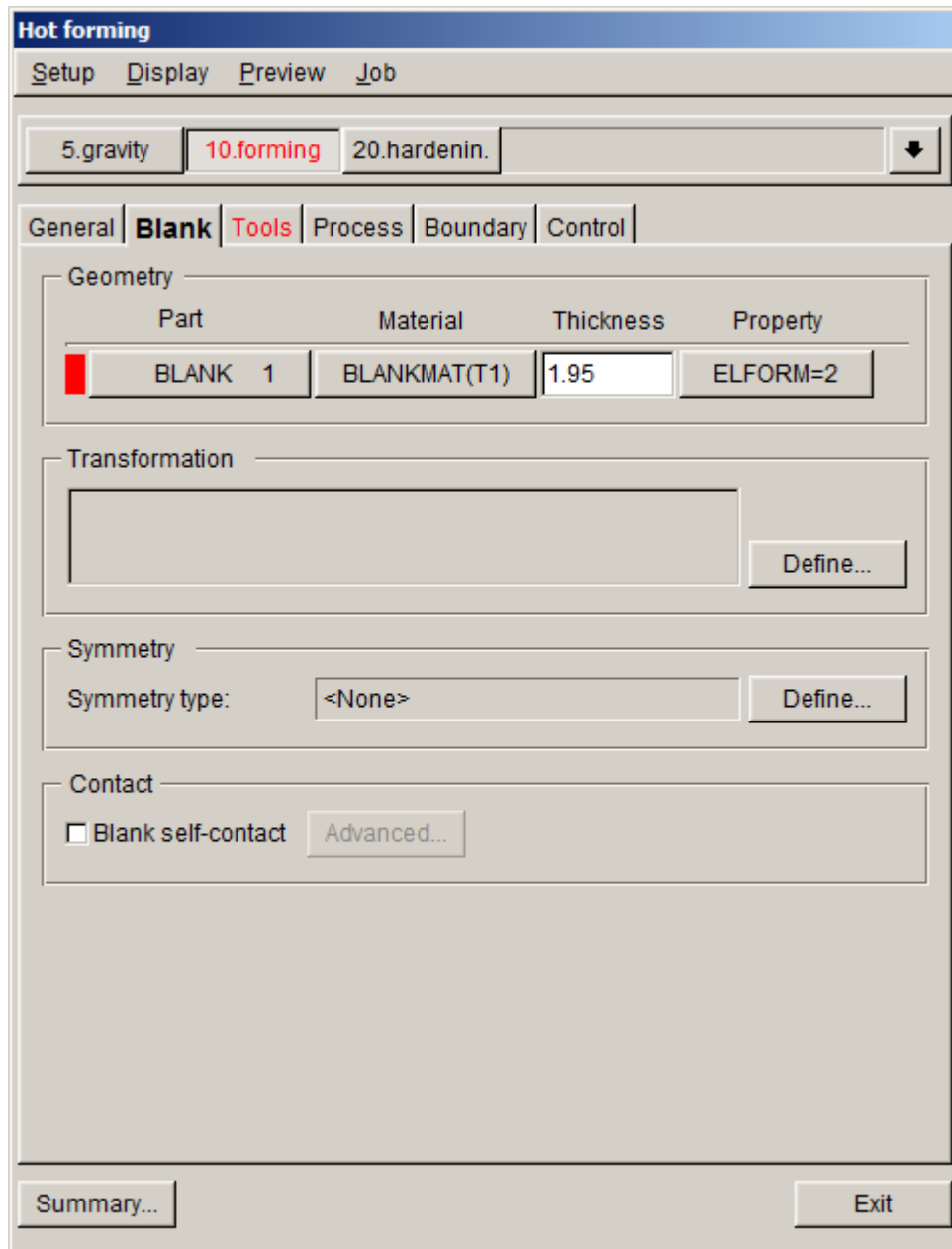


Figure 1.10 Blank definition interface

IV. Blank Material and Property Definition

In the hot forming analysis, both the structural material and thermal material need to be defined. The structural materials include MAT 106 and MAT 244.

Click the button under the material to display the material dialog box illustrated in Figure 1.11. Click the Import button to import the MAT244 material file blankmat.mat.

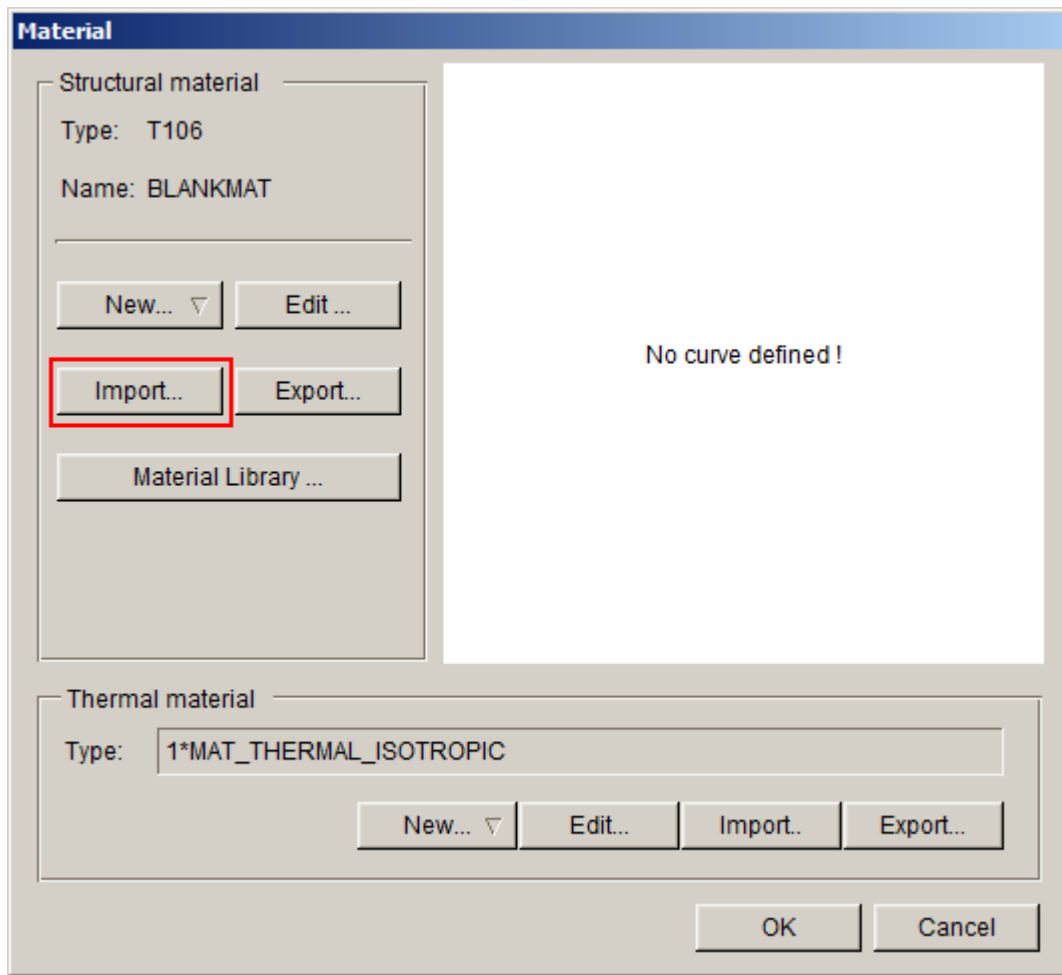


Figure 1.11 Structural material definition

Note: Users can create the material defined by them by selecting the material type, or edit the selected material.

Click the Import button to import the thermal material file blankmat_hot.mat, as illustrated in Figure 1.12.

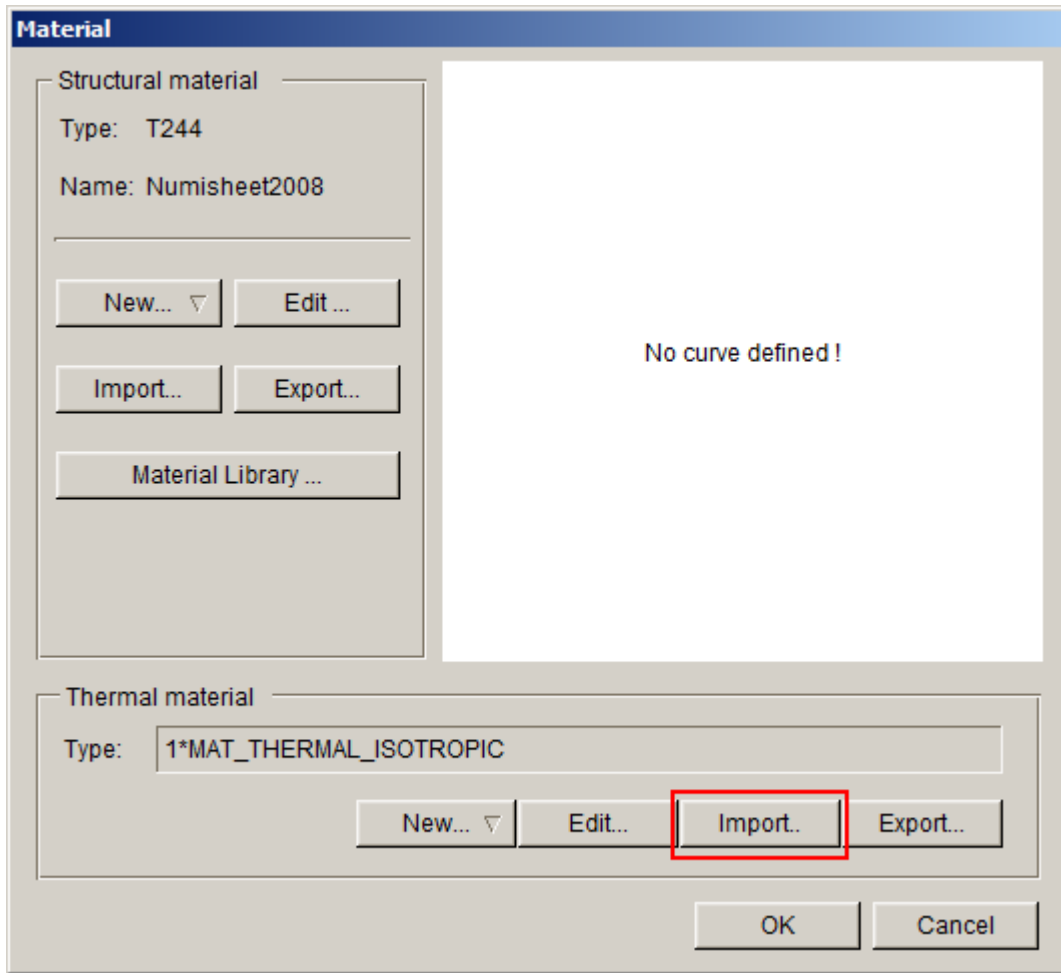


Figure 1.12 Thermal material definition

V. Tools Definition

1. Enter Tools interface, and click the red **Tools** tab to display the **Tool Definition** interface.
2. Click  icon from the **Icon** bar, and turn off the **BLANK** part. Click BLANK button to turn off the blank part.
3. Click OK to exit ON/OFF dialog box of the part. The program defines three default tools: die, punch and binder located at the left of tools interface. The user can define the tools one by one.
4. Enter the **die** interface, and select **die** located at the left of tools list, then select **Define geometry...** button to define die, as illustrated in Figure 1.13.
5. The Tool Preparation dialog box is displayed. Click **Mesh→Organize→Define tool** icon in the preparation dialog box illustrated in Figure 1.14.

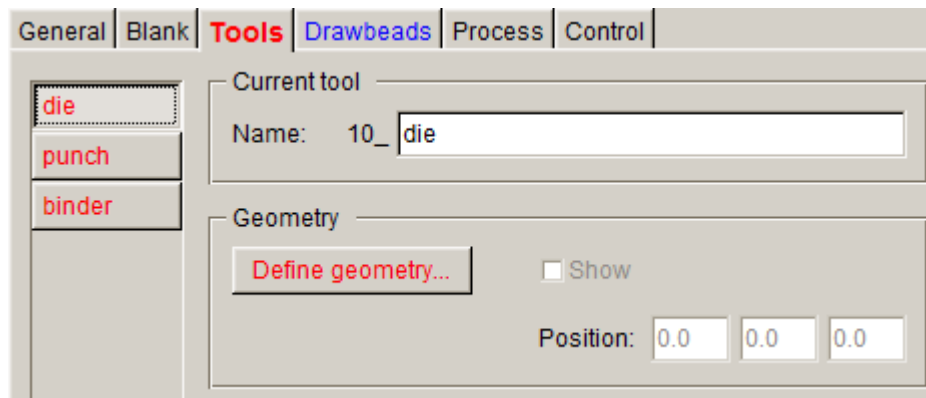


Figure 1.13 Die definition

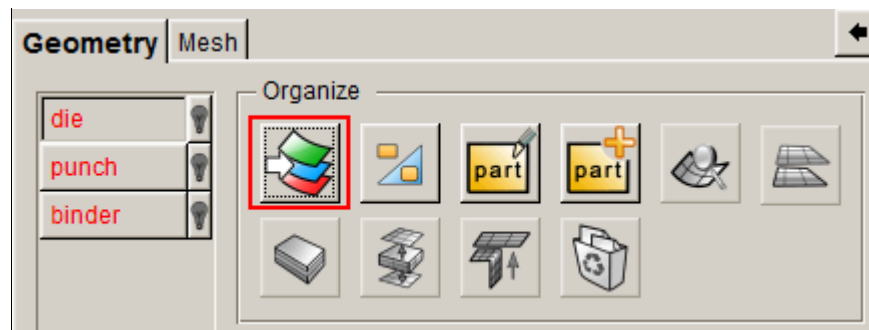


Figure 1.14 Preparation

6. The Define Geometry dialog box is displayed. Click the Add part... button, as illustrated in Figure 1.15.
7. Select the DIE button from the popped Select Part interface. See Figure 1.16.

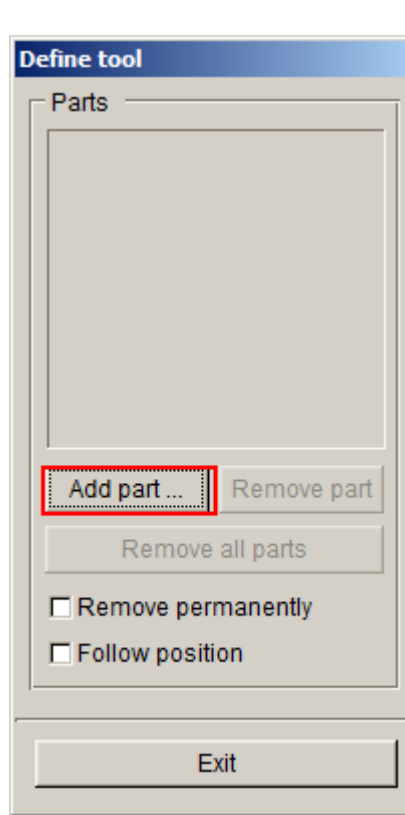


Figure 1.15 Tool definition

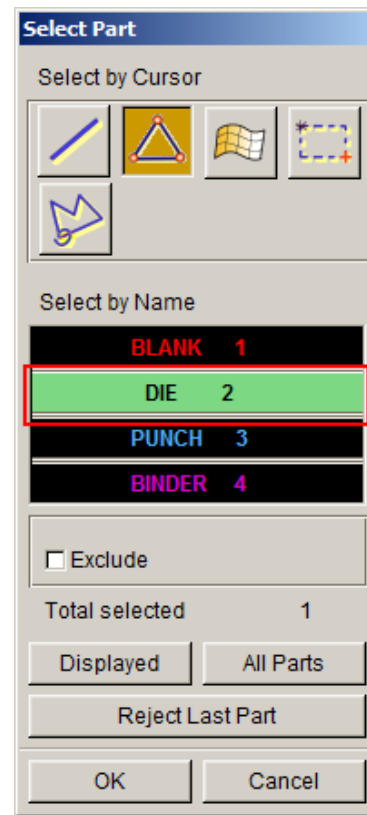


Figure 1.16 Select Part

8. Click OK to return to Define geometry interface, and now the die part has been added to part lists. See Figure 1.17.

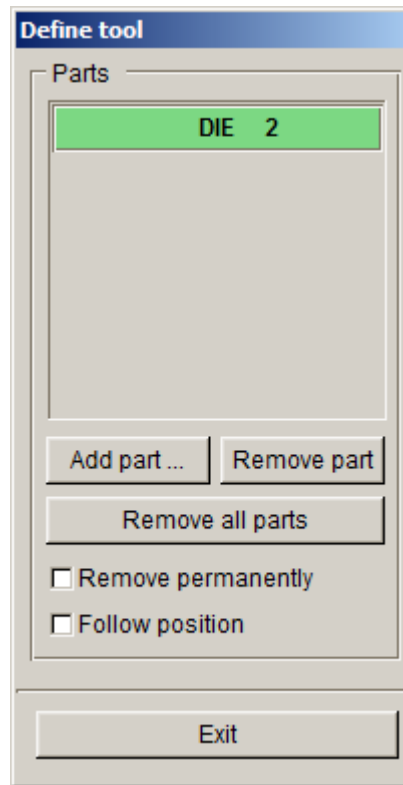


Figure 1.17 Tool definition

9. Click the **Exit** button to return to tool definition interface. Now, the die has been defined and the font color of **die** is changed from RED to BLACK. See Figure 1.18.

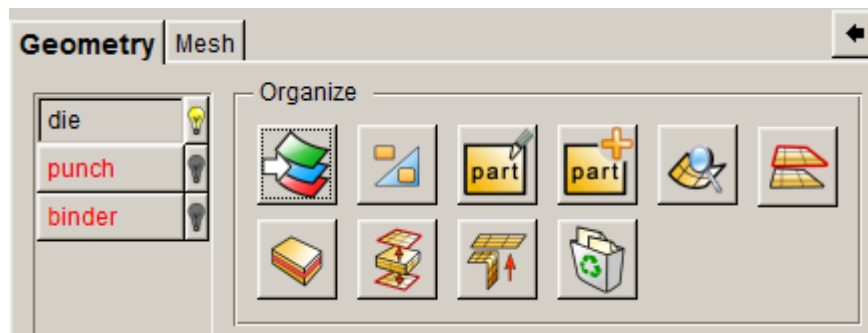


Figure 1.18 Die definition interface

10. Select **punch** located at the left side of tools list to display **punch** definition interface.
11. Finish the definition for punch and binder by referring to Step 5-Step 9.
12. Click the **Exit** button to exit the **Tool Preparation** dialog box.

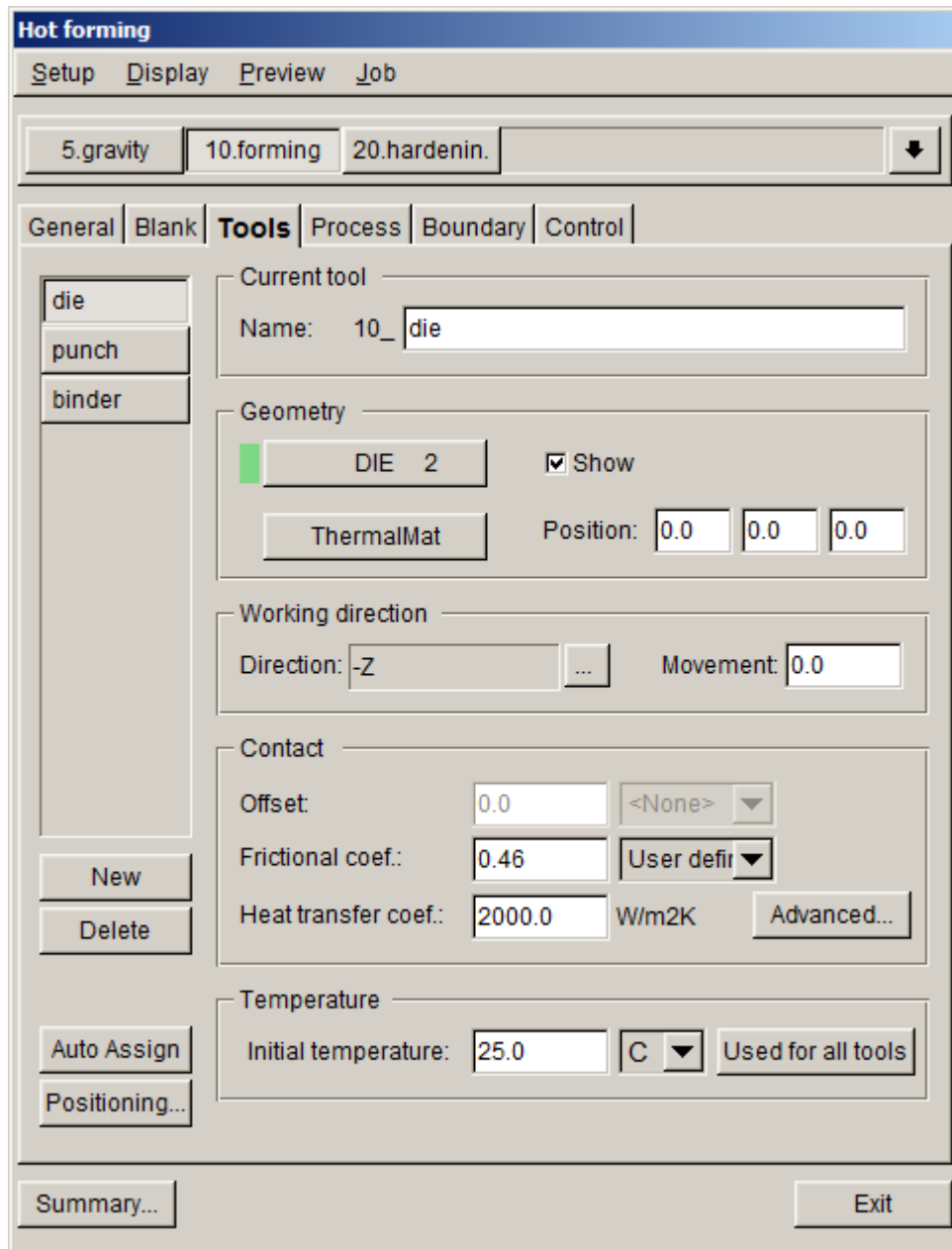


Figure 1.19 Tool definition interface

VI. Tools Material Definition

In the hot forming analysis, both the structural material and thermal material of the blank, and the thermal material of the tool need to be defined. Dynaform allows the user to same or different materials for the tool.

1. Click the ThermalMat button, as illustrated in Figure 1.20.

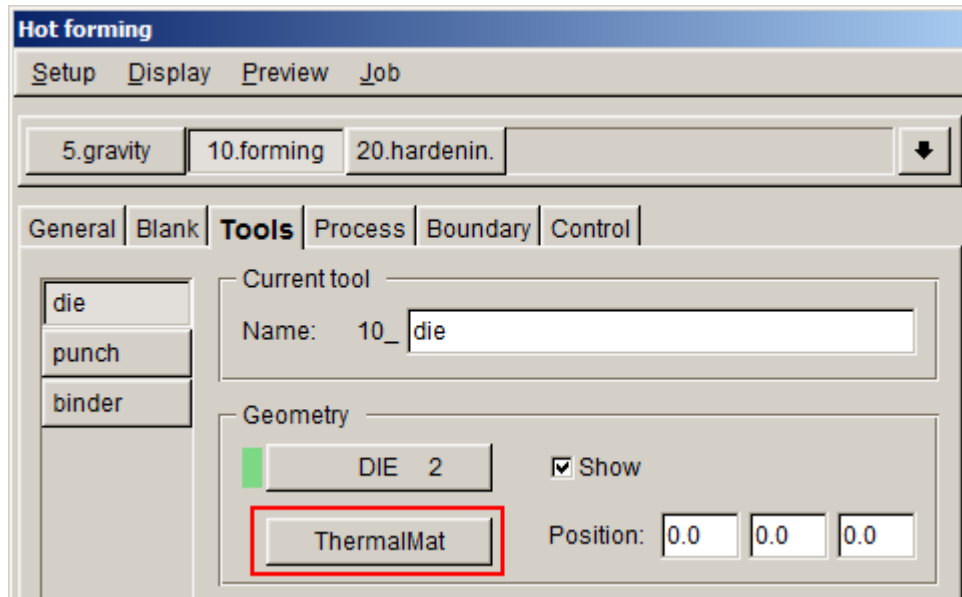


Figure 1.20 Thermal material definition of tools

2. Toggle on the Used for all tools option, as illustrated in Figure 1.21.
3. Click the Import button to import the tool material: toolmat_hot.mat.
4. Click the OK button to exit the thermal material dialog box.

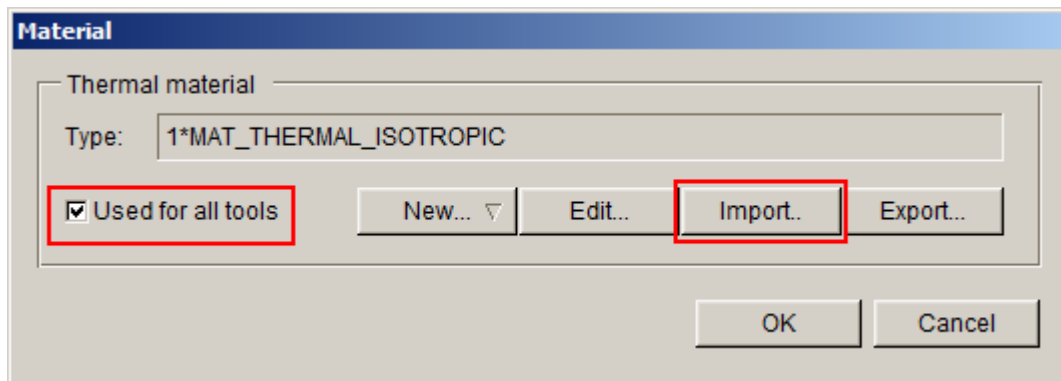


Figure 1.21 Thermal material dialog box

VII. Tools Positioning

After defining all tools, the user needs to position the relative position of tools. The tool positioning operation must be carried out each time when the user finishes the tool definition. Otherwise, the user may not obtain correct result. In addition, the tool position is related to the working direction of each tool. Therefore, you need to carefully check the working direction prior to positioning the tools.

1. Click the **Positioning** button in the Tools tab to display the Positioning dialog box.
2. The program will automatically locate other tools and the blank with binder as the reference position. The tools and the blank will be moved to a new position and the corresponding distance values will display in each edit box, as illustrated in Figure 1.22.
3. Click the Reset button to restore the original position of the blank and tool.
4. Click the **On lowers** button to position the blank and tool based on the lowers, as illustrated in Figure 1.23.

Positioning

Blank

Position: On:

Tools

Name	Movement	On
die	0.0	Blank v
punch	16.0	Blank v
binder	0.0 C	

Preference

Gap between blank and tool: ☒ Default

Gap between tools:

Vertical mesh tolerance:

☐ In the normal direction of tool surface

☒ Round off ☒ Auto position

Figure 1.22 Tool positioning dialog box

Positioning

Blank

Position: On:

Tools

Name	Movement	On
die	0.0	Blank v
punch	0.0	
binder	0.0 C	

Preference

Gap between blank and tool: ☒ Default

Gap between tools:

Vertical mesh tolerance:

☐ In the normal direction of tool surface

☒ Round off ☒ Auto position

Figure 1.23 Modify datum reference

5. Click  icon on the **Icon** bar to display the relative position of tools and blank(s), as shown in Figure 1.24.

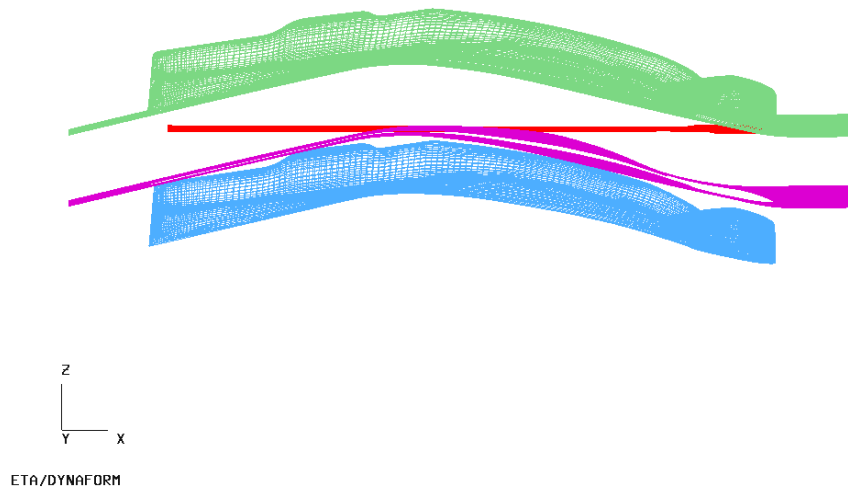


Figure 1.24 The relative position of tools and blank after positioning

6. Click **OK** button to save the current position of tools and blank(s), and return to the Hot Forming interface.

*Note: In the hot forming, after the user has performed the positioning operation to the tool, the relative position of tools and blanks is displayed on the screen. However, you may follow by clicking the **Reset** button in the Positioning dialog box to set the tools and blank(s) back to its original position.*

Now, all Tools have been positioned. The user can set up next process. In Hot Forming application, the definition and positioning of blank(s) and tools and definition of process are not required in strict order. Therefore, you can randomly modify each operation. However, an experiential engineer should have a good habit, so we suggest that the user should set up the tools and blanks step by step.

VIII. Process Definition

Process definition is helpful to setting up process numbers, time of every process, and condition of every tool and so on. The user can click **Process** of the main interface, and enter process interface. When defining a new setting, the user only need select a template. Then the program will automatically add some essential processes. To typical program, these processes need not amend or amend a little. In this way, the user can reduce setting time.

We select the **Single Action** template before, so the forming brings two default processes, one is binder process, and the other is draw process.

1. Select closing process from the list located at left side of the interface as the current process. Modify the default setting of closing stage, as shown in Figure 1.25(a).
2. Select drawing process from the list located at left side of the interface as the current process. Modify the default setting of drawing stage, as shown in Figure 1.25(b).

Hot forming

Setup Display Preview Job

5.gravity 10.forming 20.hardenin. ↓

General Blank Tools **Process** Boundary Control

closing
drawing

Current step
Name: 10_ closing ☐ Hydro

Tool control

Tools	Action & Value		
die	Velocity ▼	2000.0	Trapezoid ▼ ...
punch	Stationary ▼		
binder	Stationary ▼		

Duration

Type: Travel ▼

Tool: die ▼ Displacement: 118.0

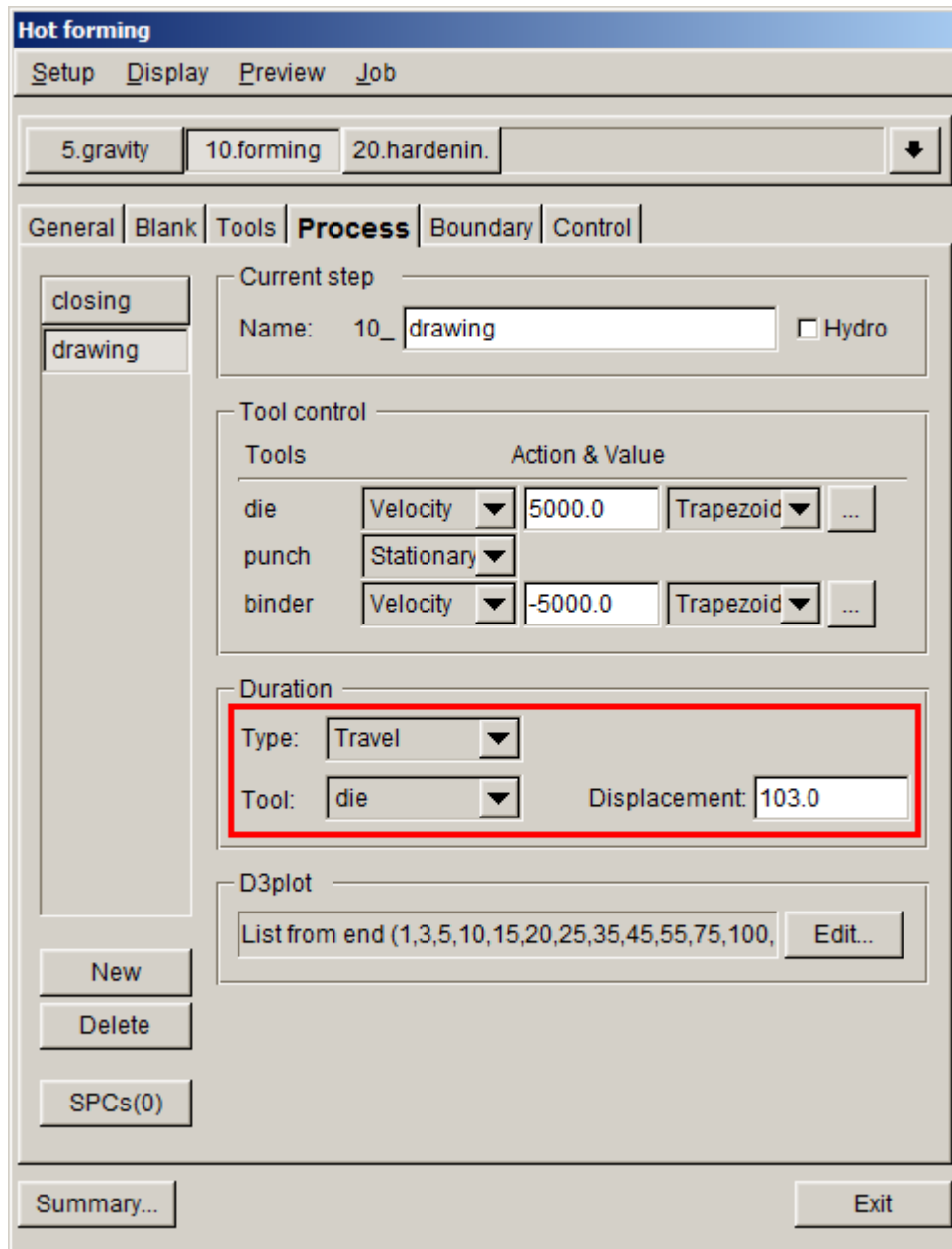
D3plot

Total number = 5 Edit...

New
Delete
SPCs(0)

Summary... Exit

a) Closing process interface



b) Drawing process interface

Figure 1.25 Process definition

IX. Control Parameters

In Control interface, use the default parameters suggested by the program. For different model sizes and mesh sizes, we suggest the following operations.

Time step size and adaptive steps

1. Click  button followed by Time step size (DT2MS), and the program will automatically calculate the time step sizes of the minimum ten elements and the average time step size of all elements. As the method that adopts the minimum time step size has a

good computing stability, but it will take much longer to calculate. Therefore, the user can enter the proper time step size with reference to the above values. Click OK button to accept the calculated time step size for this example, as illustrated in Figure 1.26.

- Click  button behind Thermal Time step (TMAX), and the program will automatically calculate the recommended thermal time step according to the tool movement velocity.

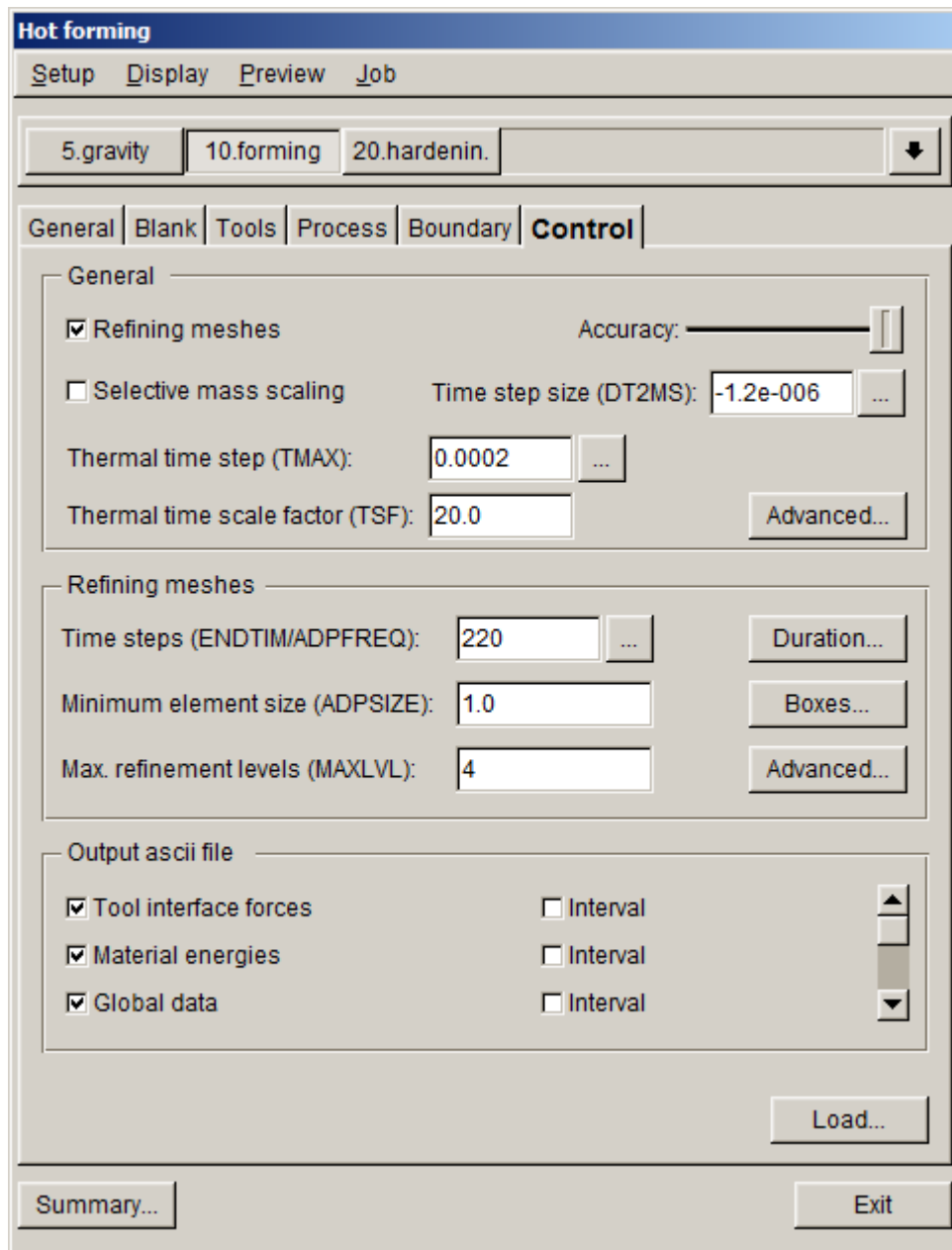


Figure 1.26 Control parameters

X. Animation

Now, the user can display model setting by animation in order to check the movement condition of the tools.

1. Click **Preview→Animation** button from the menu bar, as illustrated in Figure 1.27.

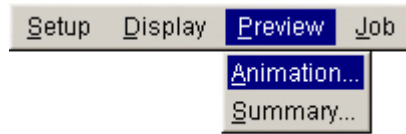


Figure 1.27 Animation menu

2. Click the **Play** button to show the animation of tool movement according to the defined motion curve.
3. The user can select the **Individual Frames** option from the Animate dialog box illustrated in Figure 1.28 to display the incremental tool movement. Click  icon to display the tool movement step by step.

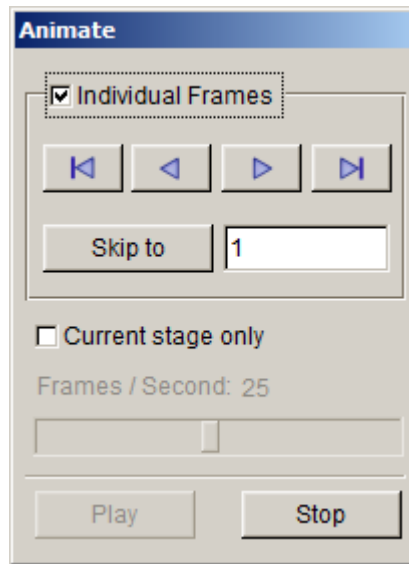


Figure 1.28 Animate dialog box

4. It will display the positions of all tools relative to their original states on the screen. See Figure 1.29.
5. Click **Exit** button to return to the Hot Forming Setup interface.
6. Click **Setup→Save** to save the database.

```
FRAME = 6      TIME = 6.057472BE-002 STAGE = forming(closing)
10_DIE      POSI = 119.149457
10_PUNCH    POSI = 0.000000
10_BINDE    POSI = 0.000000
```

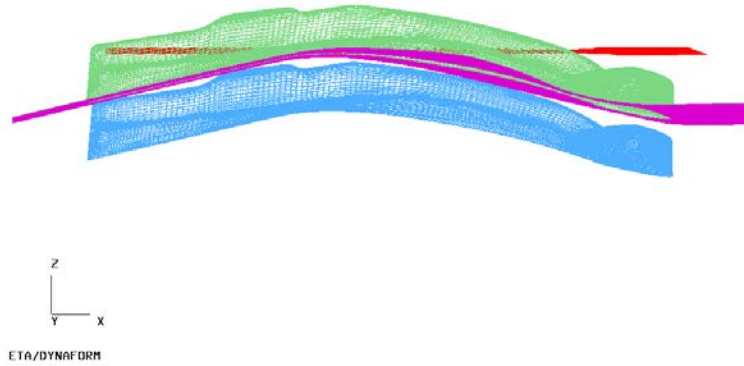


Figure 1.29 Animation of tool movement

XI. Gravity Loading

A certain softening occurs to the blank in the hot analysis; therefore, the gravity loading analysis is suggested to conduct generally. It cites the blank and tool of the forming stage in the gravity loading, so the user does not need to re-define the blank and tool. The temperature distribution and the stress-strain information after the gravity loading will automatically transfer to the forming stage.

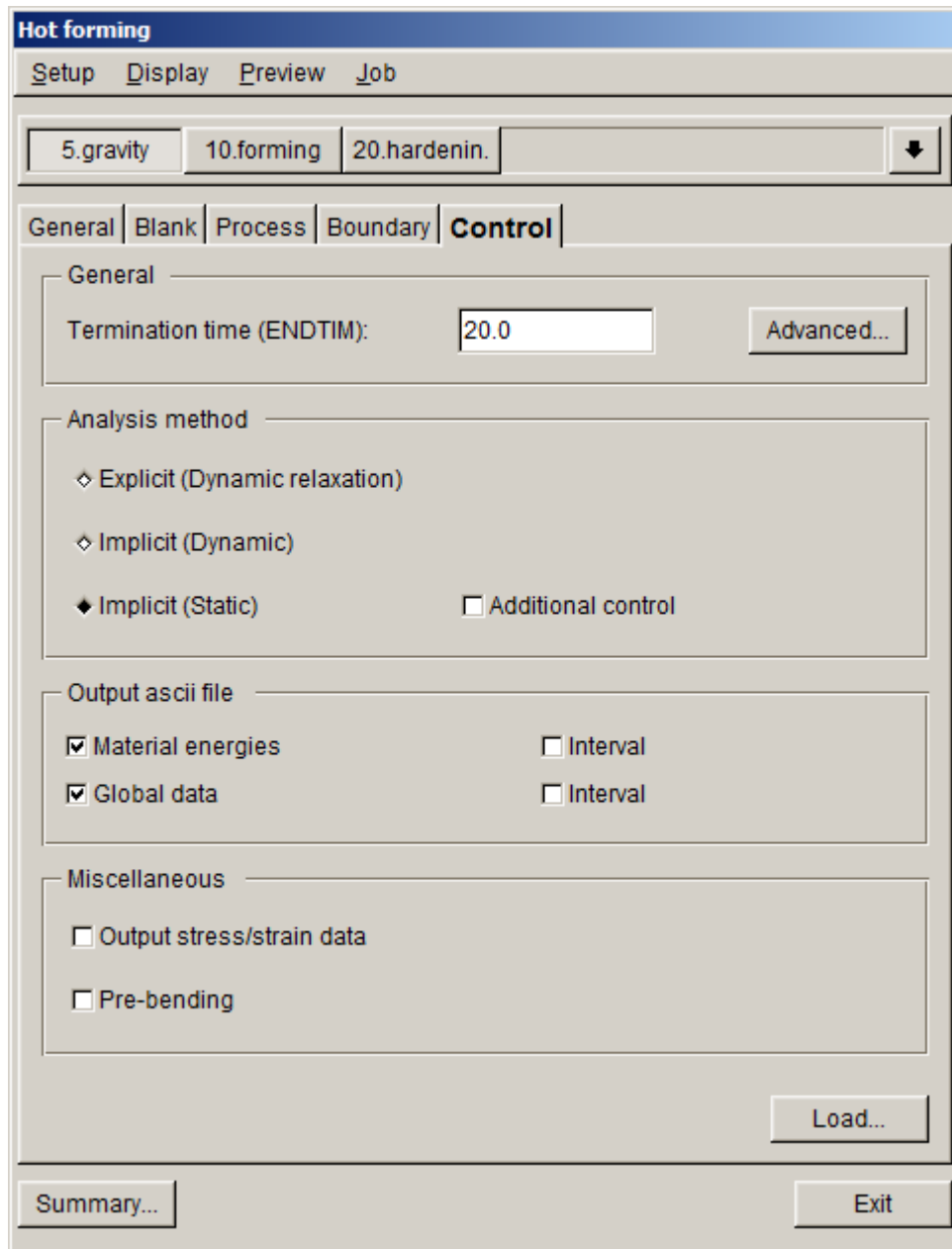


Figure 1.30 Gravity Loading

In Control interface, click the Advanced button to modify the following parameters, as illustrated in Figure 1.31.

Thermal control	
HOURLASS(IHQ)	4
SHELL(ESORT)	1
SHELL(THEORY)	2
SHELL(BWC)	2
SHELL(PROJ)	0
CONTACT(SLSFAC)	0.01
CONTACT(ISLCHK)	2
CONTACT(SHLTHK)	1
CONTACT(PENOPT)	4
CONTACT(THKCHG)	0
CONTACT(XPENE)	4.0
OUTPUT(NPOPT)	1
SOLVER(ATYPE)	1
SOLVER(PTYPE)	2
SOLVER(SOLVER)	3
SOLVER(FWORK)	1e-010
TIMESTEP(ITS)	0.5
TIMESTEP(TMIN)	0.01
TIMESTEP(TMAX)	5.0
TIMESTEP(DTEMP)	25.0

OK Default Cancel

Figure 1.31 Thermal control parameter

XII. Press Hardening

The blank, tool temperature and stress-strain information in the forming stage will be automatically transferred to the press hardening stage. In this stage, the tool is located in the end position. By default, the force for die is 40 tons, and duration time is 5 seconds, as illustrated in Figure 1.32.

Hot forming

Setup Display Preview Job

5.gravity 10.forming 20.hardenin. ↓

General Blank **Process** Boundary Control

hardening

Current step

Name: 20_ hardening ☐ Hydro

Tool control

Tools	Action & Value		
die	Force ▼	400000.0	Constant ▼ U
punch	Stationary ▼		
binder	Stationary ▼		

Duration

Type: Time ▼

Time: 0.05

D3plot

Total number = 5 Edit...

New

Delete

SPCs(0)

Summary... Exit

Figure 1.32 Press Hardening process definition

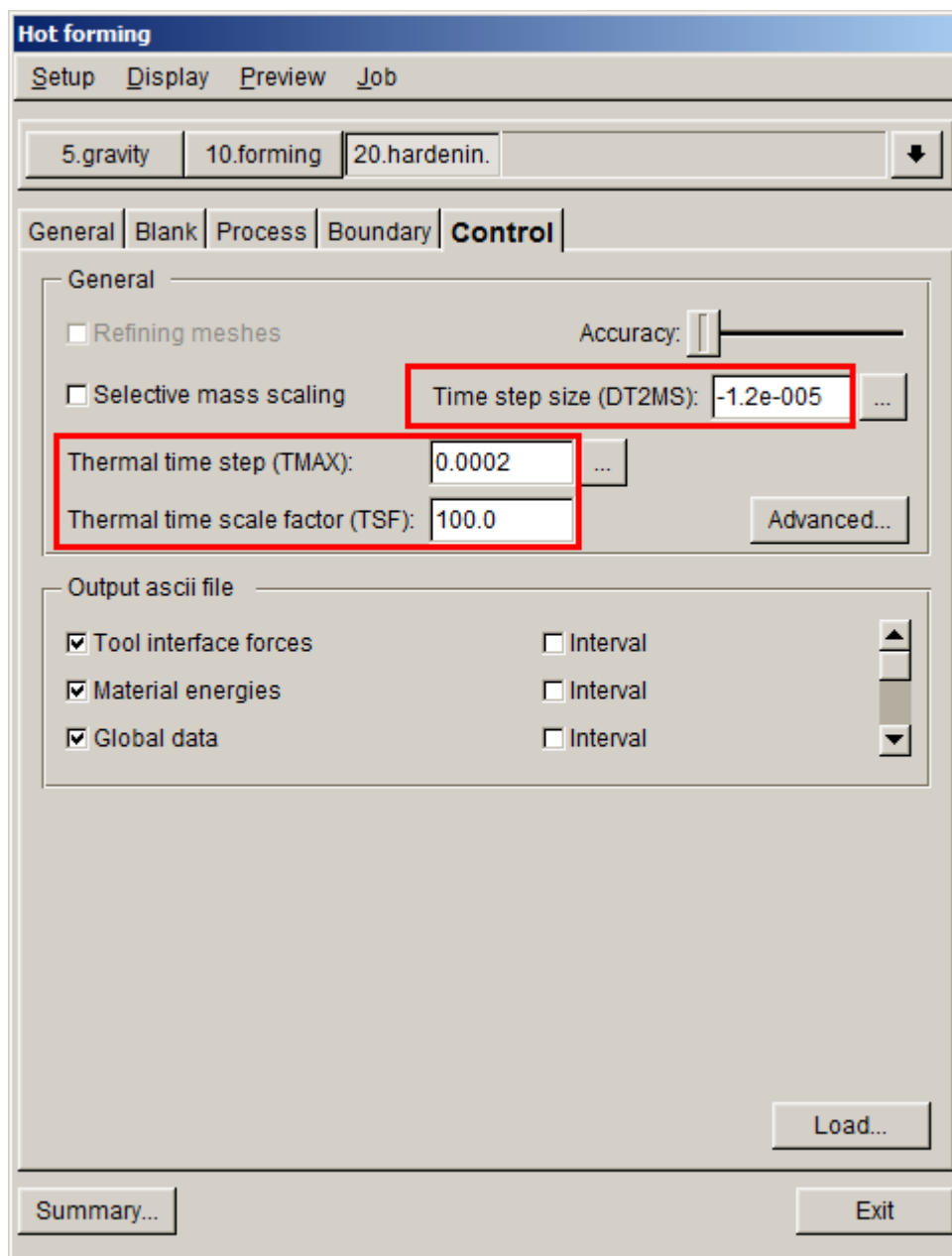


Figure 1.33 Press Hardening control parameters

XIII. Submit Job

After defining all the setup, you may submit jobs of the current setting for calculation.

1. Click **Job→Job Submitter** in the menu bar, as illustrated in Figure 1.34.
2. The program pops up a Job Options dialog box illustrated in Figure 1.35. LS-DYNA double precision solver is adopted for calculation in the hot forming analysis.
3. Click OK button to complete the option setup.

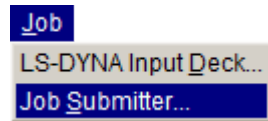


Figure 1.34 Job menu

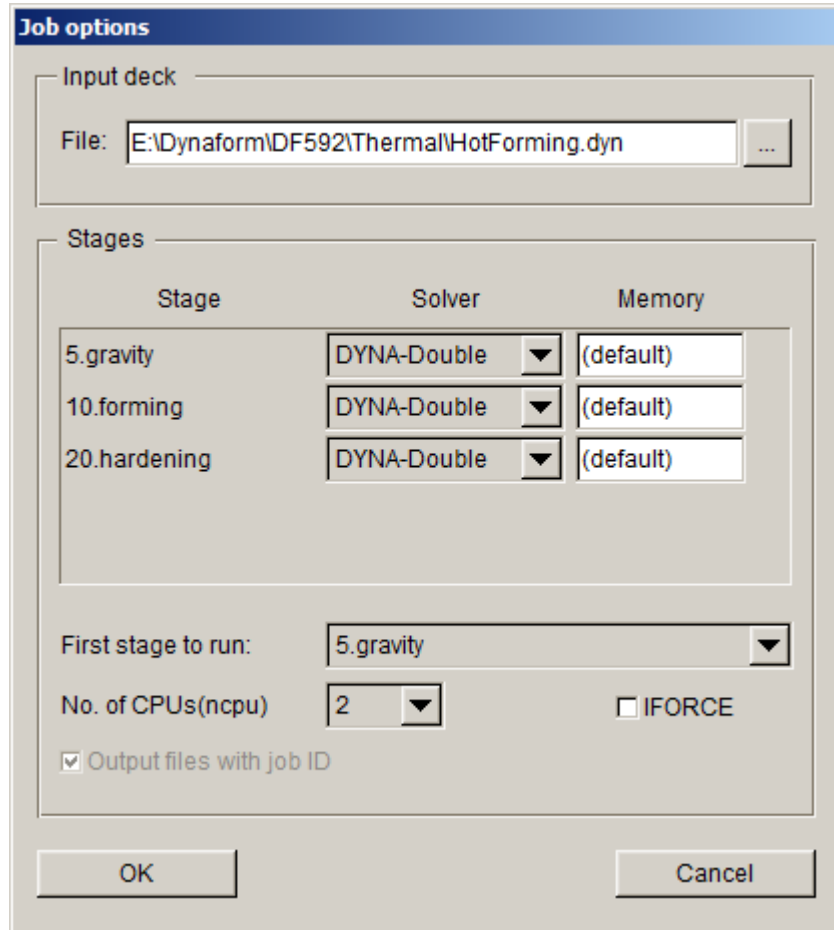


Figure 1.35 Job options

4. The current job has already been submitted to the Job submitter for unified management. If there is not any other job in the Job submitter list, the LS-DYNA window is displayed.

POST PROCESSING (with eta/POST)

The eta/POST reads and processes all the available data in the **d3plot**. In addition to the undeformed model data, the **d3plot** file also contains all result data generated by LS-DYNA (stress, strain, time history data, deformation, etc.).

I. Reading the Results File into the Post Processor

To execute eta/POST, click PostProcess from eta/DYNAFORM menu bar illustrated in Figure 1.36. The default path for eta/POST is C:\Program Files\ETA\DYNAFORM5.9.2.1. In this directory, you can double click the executable file, EtaPostProcessor.exe to open eta/POST GUI illustrated in Figure 1.37. The eta/POST can also be accessed from the programs listing under the start menu of eta/DYNAFORM.

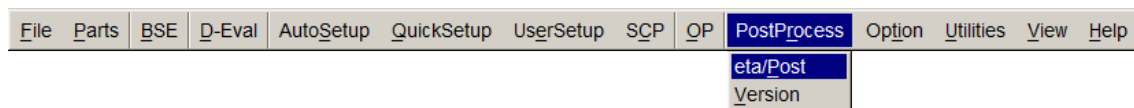


Figure 1.36 Post Process menu

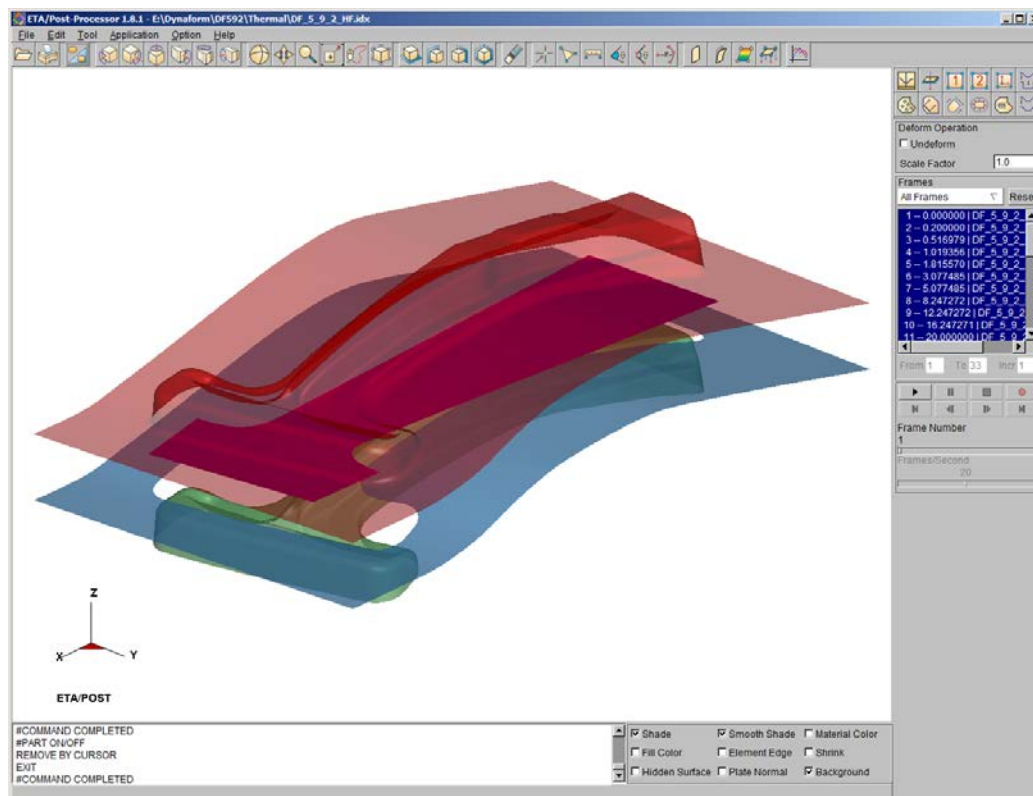


Figure 1.37 Post process GUI

1. Select the menu **File → Open** in eta/POST to display the Open dialog box.
2. Select ETA Multiple stage file (*.idx) as the file type, as illustrated in Figure 1.39.
3. Select the HotForming.idx file.
4. Click the **Open** button to open the calculation result file.

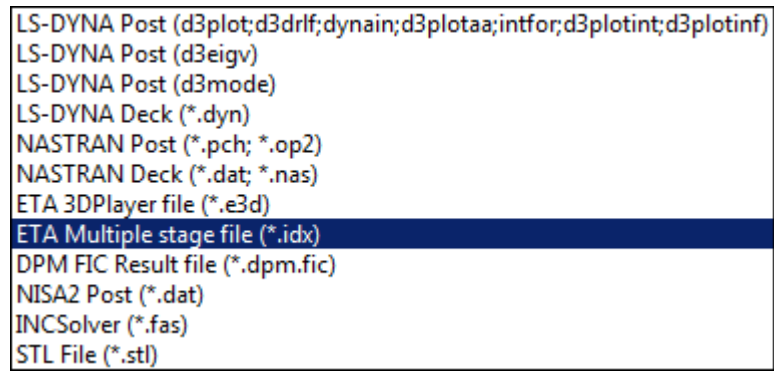


Figure 1.38 File type list

- The user is allowed to perform the post processing to the result file, and the operation toolbars are illustrated in Figure 1.39.



Figure 1.39 Tool bar of eta/POST

II. Deformation

- The default **Plot State** is **Deformation**. In the drop-down menu of **Frame**, choose **Single Frame** option, as illustrated in Figure 1.40.

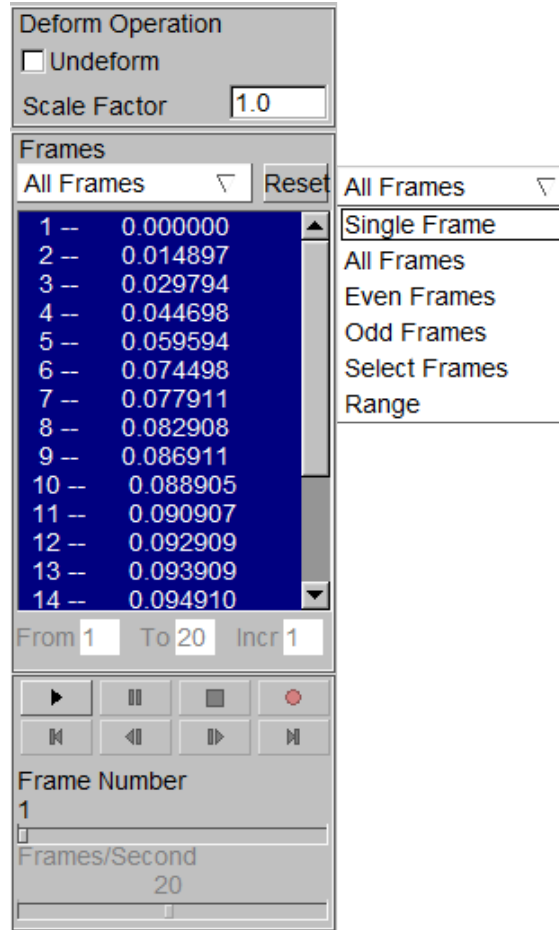


Figure 1.40 Deformation dialog box

- Since it is difficult to see the Blank with all of the other tools being displayed, you can hide all the other tools by clicking the  icon from **Icon bar**.
- In Part Operation dialog, close all the tools except BLANK part. The color of all the closed parts turns to white.
- Click **Exit** to exit the operation.
- The user may also change the displayed model using the view manipulation icons on the **Icon bar** illustrated in Figure 1.41.



Figure 1.41 View manipulation icons

III. Thinning

- Select  icon from the tool bar.
- Select THINNING from the drop-down, as illustrated in Figure 1.42.
- Click the **Play** button to animate the thinning contour.

4. Move the **Frame/Second** slider to set the animation velocity.
5. Click the **Stop** button to stop the animation.

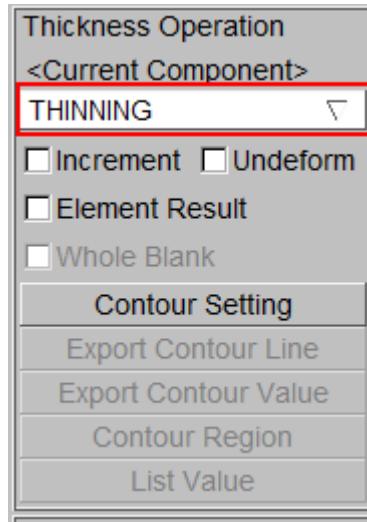


Figure 1.42 Thickness operation dialog box

IV. View Temperature Distribution

For the hot forming analysis, the user more concerns about the temperature distribution of the blank and tool.

1. Click  icon from the tool bar.
2. Select Temperature from the Contour Plot/Animation drop-down.
3. Select Temperature from the Current Component drop-down, as illustrated in Figure 1.43.

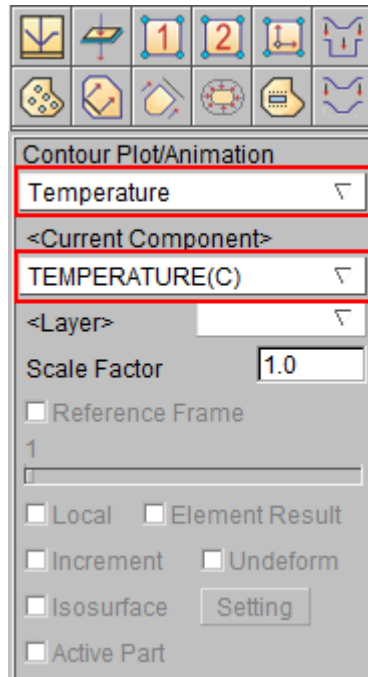


Figure 1.43 Temperature display

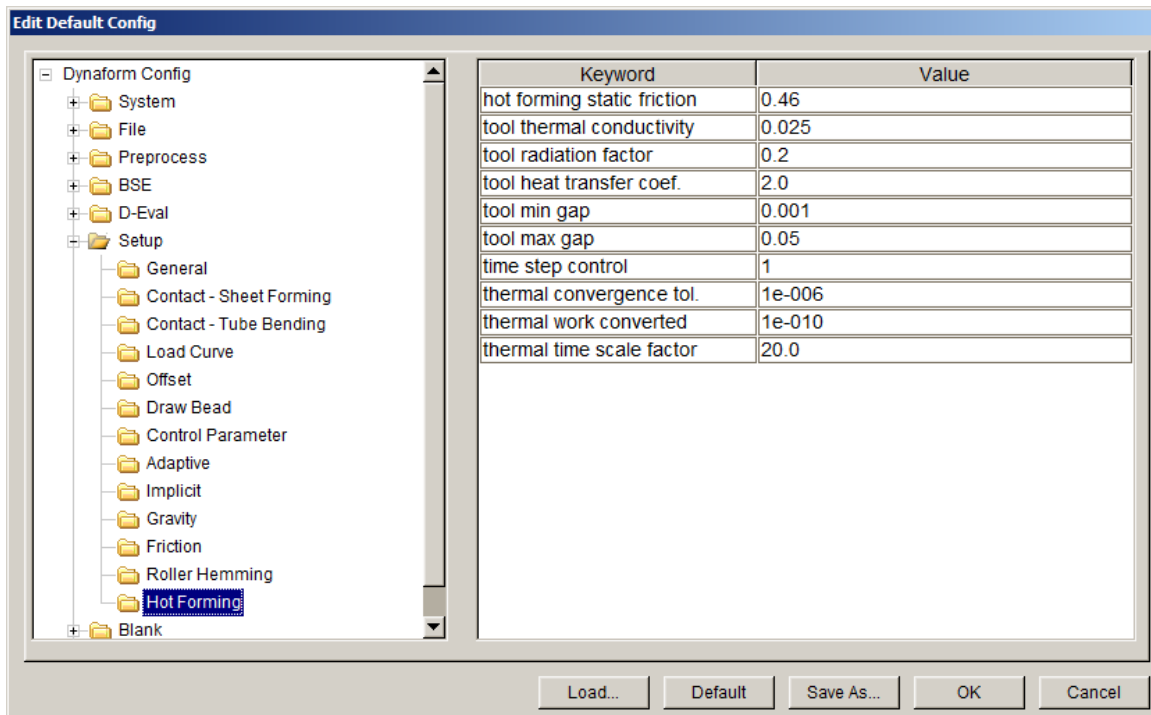
Note: In addition to the above-mentioned functions, eta/POST also provides the functions such as Major/Minor Strain, FLD, Vector, Circular Grid and Blank Tool Distance. The user can refer to eta/POST User's Manual for detailed description about these functions.

MORE ABOUT eta/DYNAFORM 5.9.2.1

The eta/DYNAFORM 5.9.2.1 configuration file is located under both the installation directory and My Documents directory. Inside the eta/DYNAFORM 5.9.2.1 installation directory, there is a file called dynaformdefault.config. Many key default parameters are included in this file. When the user opens eta/DYNAFORM 5.9.2.1 for the first time, the program will automatically create a file called dynaformdefault.config under My Documents directory. The configuration file locates in the “My Documents” directory of the current user’s PC, for example, C:\Documents and Settings\Andy\My Documents\ETA\Dynaform5.9.2.1 (Windows XP), C:\Users\Andy\My Documents\ETA\Dynaform5.9.2.1 (Windows 7). The user can open this file and change these default parameters, or click **Option→Edit Default Config** from Menu bar to edit these default parameters in the dialog box. The modified default values in the dialog box will be saved in the configuration file under My Documents directory. The configuration file under My Documents directory will take precedence over the one under the installation directory.

For UNIX/Linux users, the dynaformdefault.config file is located under both the installation directory and the user’s home directory (/home/Andy/ETA/Dynaform5.9.2.1). The dynaformdefault.config file located under the installation directory will take precedence over the one located in the user home directory.

The default parameters for the hot forming are illustrated in the following figure.



Default parameters in hot forming

CONCLUSION

This concludes the training guide's basic overview of eta/DYNAFORM 5.9.2.1. This manual is meant to give you a basic understanding of finite element modeling for forming analysis, as well as displaying the forming results. It is by no means an extensive study of the simulation techniques and capabilities of eta/DYNAFORM. For more detailed study of eta/DYNAFORM, you are urged to attend an eta/DYNAFORM training session.

Please refer to the *eta/DYNAFORM User's Manual* and the *LS-DYNA Keyword User's Manual* for detailed description of individual functions and analysis settings.